



Gunthard Kraus, DG8GB

An interesting program

APLAC

1.

Brief description

Name: APLAC

Area of application:

APLAC is a universal program for analysing and synthesising every conceivable circuit or system for telecommunications, electronics and physics in a specific time or frequency range. This includes EM simulators, using the FDTD method.

Most important positive characteristics:

There is practically nothing that cannot be investigated using APLAC.

Just to show the application options, the parts list contains not only all the electronics components previously known, but also models for mechanical oscillation systems and shock absorbers. And the manual even provides an example of the simulation of RF energy absorption in the human skull when a telephone call is made on a mobile phone.

What I liked:

All manuals are available for downloading at no cost and constitute complete textbooks. All known microwave components and models of irregularities (from the microstrip bend through the interdig-

ital capacitor and length coupler through to the circulator) are also released in the free student version. An unbelievably comprehensive library relating to functions and options, not just in electronics, but also in mathematics and physics.

A good, easy to operate editor for “drawing” wiring diagrams.

For experienced users and if all the effects playing a role are included, the simulation results tally precisely with reality.

Problem areas:

The program is a universal tool and is thus not equipped with easy to operate button control (like ANSOFT SERENADE). It operates in its own “APLAC language”, which initially requires the operator to set up several “control objects”. These are then combined to form a “simulation file” and executed. In plain language this requires a lot of writing and separate programming for the solution description, the sweep co-ordinates and for displaying the results in diagrams, text files or tables. In principle, this all represents a mixture of PSPICE and ARRL radio designer command lines, together with elements of the programming language “C”.

File type and file size:

A download of approximately 45MB is required for all program sections and



manuals. Following installation, that gives a Windows file of approximately 65MB.

No problems could be detected during operation using WIN 95/98/ME/ 2000/NT or fast computers.

Bugs or serious miscalculations detected:

Only insignificant little details or idiosyncrasies (e.g. in the graphics structure or the print out), which could have been clarified through the "APLAC Internet hotline" using email. One pleasant thing is that any query to this hotline (whether it is from a student or from an industrial firm) is treated in an impartial and friendly manner, and is handled competently and free of charge.

Desirable improvements:

Intensive further development of what has already been started, to make inputting using wizards (small additional programs with graphics user guides) considerably easier.

Expansion of permissible simulation content in free student version (see also under "Limitations").

User friendliness:

As already pointed out, a very intensive familiarisation phase is required before the number of error messages following the start of a simulation is reduced. Perfect control is not achieved until some time has elapsed and then only with continuous usage. Nevertheless, the operator has more to do than for similar products.

Help:

Very comprehensive online PDF manuals in 7 volumes, each with approximately 1,800 pages. Many examples of circuit and EM simulation as files on CD, together with discussion in the manual. Very friendly support, free of charge, obtainable via Internet or email.

Limitations:

The free student version restricts the number of components to a maximum of 12, although resistors, capacitors and coils are not counted. In addition, the main memory in use during simulation must not exceed a specific size, or else the processing is interrupted.

One way out for experienced users in such a case is to request the transfer to a 45 day full test version free of charge.

Acquisition options:

Download student version from <http://www.aplac.com> or through "Internet Treasure Trove CDs" from VHF Communications.

2.

A brief overall view

Professor Marti Valtonen developed APLAC (Analysis Program for Linear Active Circuits) back in 1972 in its basic form. Since then, it has grown and flourished in the fertile soil of Helsinki University (and no doubt within the countrys NOKIA culture as well). As already stated, the basic concept clearly goes beyond the limits of the rival products, which mainly specialise in certain specific simulation areas and then look for business from customers with sophisticated operating systems.

APLAC, on the other hand, is designed to handle every question, and brings together everything conceivable or known in the world regarding simulation tools. This covers the PSPICE module for linear circuit simulation through the harmonic balanced simulator to non linear large signal analyses, from the communications system simulator through the regulation of noise sidebands in oscillators right through to the EM simulation of microstrip antennas or RF absorption in the human skull. It is only

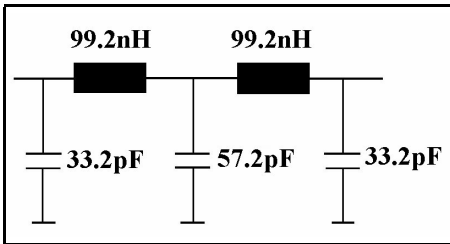


Fig 1: The results of a filter program for a 110MHz Chebyshev low pass filter.

after an intensive study of the comprehensive manuals that you discover everything you can do with this system.

Universality like this naturally has to be paid for, in this case with low user friendliness. This means that you have to assemble everything yourself and you have to tailor everything to the application in question. For this, you have to use the systems own "APLAC language" that you may well take some time to get into your head and which you'll probably never understand completely.

Setting up the necessary simulation files is similar to C programming. On the first run through there will be any number of

error messages, but with increasing experience they can be eradicated ever more rapidly. Once this hurdle has been overcome, not only can we enjoy the precise results, but also the numerous investigation and display options very quickly console us for the relatively laborious operation.

Now some simulations to explain the right way to use the software and simultaneously demonstrate its efficiency.

2.1. First simulation example:

S Parameter of a 110MHz Chebyshev low pass filter

2.1.1. Setting up the circuit diagram

Before we can start up APLAC, the component values of the desired low pass filter are determined, using one of the free DOS filter programs from the Internet (e.g. "fds" or "faisyn20"). The following attributes should apply here:

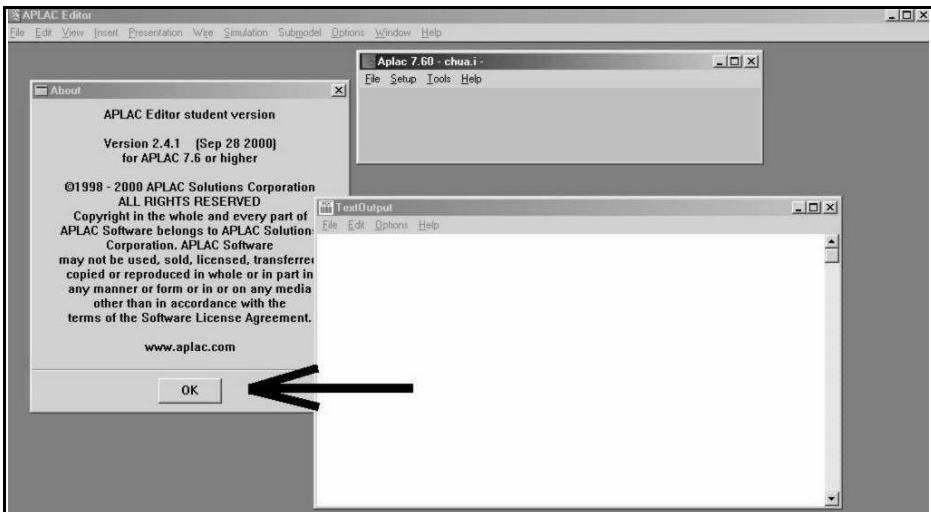


Fig 2: After starting APLAC press OK to start the editor.

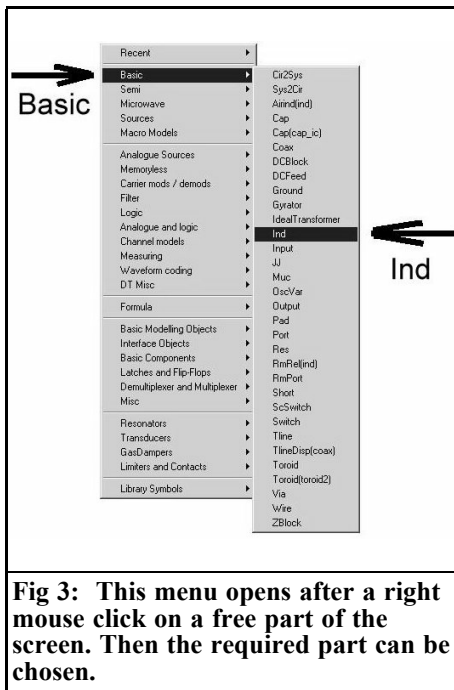


Fig 3: This menu opens after a right mouse click on a free part of the screen. Then the required part can be chosen.

- Ripple = 0.1dB.
- Ripple limiting frequency, $f_g = 110\text{MHz}$
- Low inductance Implementation
- Degree of filtration, $n = 5$
- System resistance, $Z = 50\Omega$

The result can be seen in Fig 1.

Now we start up APLAC, the screen is shown in Fig 2. This shows how the circuit diagram editor is started up. Use the white drawing surface that appears and make it full screen. We then right click the mouse somewhere in the free area of the screen. A new menu appears and we select "Basic". This calls up the list of basic components, and we can select an inductance, using the term "Ind" (Fig 3). Following confirmation, using the <ENTER> key, the coil is suspended from the cursor and is deposited in the wiring diagram. Repeat the sequence and you will have the two coils

required. Any component used is also listed under "BASIC / RECENT" and can be called up from there.

Now we call up the three capacitors under the term "Cap". However, each time they are placed they must be rotated through 90° using the key combination <CONTROL> + <R>. We also need three earth symbols, in the basic parts list the earth symbol is shown as "Ground".

Finally we double click on the graphic for each component and enter its value in the "Attributes" field. Do not forget to make the numbering of the component and its attributes visible using "Show Name" and "Show all Attributes". When a value is entered always use decimal points (American notation) and not commas!

And now for the wiring, just double left click anywhere in the free area of the wiring diagram, and a "wire coil" will suddenly be suspended from the cursor. Click on the connection of a component and the wire then uncoils to the next connection desired.

The connections here may, and should, run in curving, oblique lines over the circuit. Just left click again, and the editor itself will automatically turn the wiring into very neat right angled shapes!

For input and output, right click to call up a port via "Basic" and deposit the symbol on the input side. Then repeat on the other side for the output port. If you connect the two ports to the circuit and to earth, then you should have a wiring diagram corresponding to Fig 4. (Please always make all attributes visible after double clicking on a symbol). Looking carefully you will notice that the system impedance is pre-set to 50Ω for both ports.

To see the details better you should zoom in on the circuit to make it full screen. To do this, go into the "View" menu and select "Zoom Whole Diagram". You have now finished drawing the circuit.

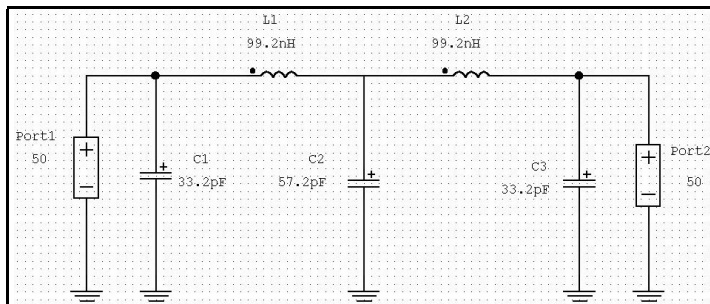


Fig 4: The completed circuit diagram in APLAC.

In addition to this, the simulated data for the low pass filter can also be written into an S parameter file in Touchstone format (with the ending `“.s2p”`). To do this, click on the output port symbol and enter the corresponding requirement

STORE “LPF110MHz.s2p” GHZ MA

for the attributes (in a new line following the 50Ω system impedance).

2.1.2. Circuit simulation

First, open the “Object List Box” (Fig 5) and double click on it. In the menu which opens, first select the setting “Isweep” and then remove it with “Delete”, because this is an “interactive Windows sweep”, that should not be used here. Replace this by a new sweep type object and type the following settings:

```
"110MHz_low_pass"
LOOP 1001 FREQ LIN 100KHZ 0.3GHZ
WINDOW=0 Y "" "dB" -50 0

...
SHOW W=0 DB S(1,1)
+ DB S(2,1)
```

Explanation:

1st line:

The term “110MHz_low_pass” is automatically used as the name of the project for each diagram and results print out and is included on the print out

2nd line:

Carry out a simulation in the frequency range and divide the range from 100kHz to 3GHz linearly into 1001 simulation steps.

3rd line:

Enter “Window=0” for the results print out set up for a plot diagram with the vertical axis numbered from -50 to 0dB.

Note:

The blank line separates the simulation section from the print out section and should therefore never be forgotten!

5th line:

Show the curves for S11 and S22 in dB in the window number zero.

Now re-open the “Presentation” menu. The sweep file has to be made visible on the screen. To do this, click on “Show Control Object”, select “Sweep” from the list and close with “OK”.

Two little things should be mentioned before starting the simulation. First, you must save the current project under a suitable name in a suitable directory. Secondly, make a final visual check of the object box list, as the sweep file must always be right at the bottom of the list (otherwise you will receive an error message which is difficult to understand). Fig 6 shows what everything should look like.

If all settings are correct, start the simulator, using the key combination <Control + S>. The results can be zoomed to full

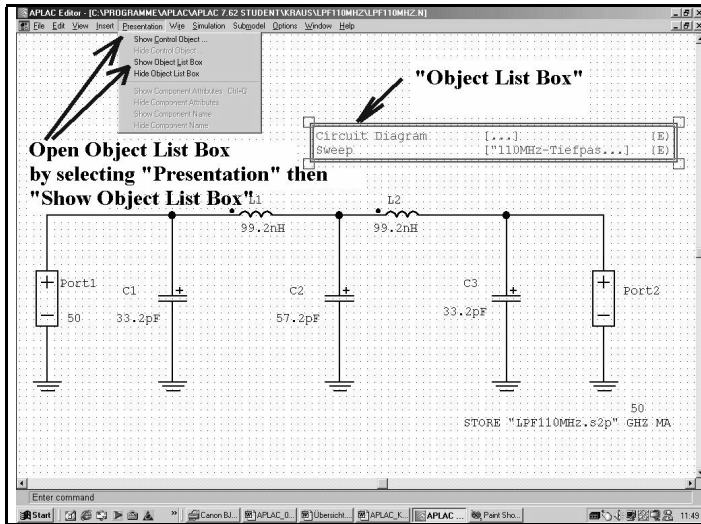


Fig 5: Opening the Object List Box to set up the required sweep object.

screen in the usual Windows manner as shown in Fig 7.

To see the Chebyshev ripples in S21 should move the cursor to the lowest value in the vertical scale, the cursor then has a small additional directional arrow. As an example, click on the value "-50 dB", a short menu opens and type "-0.4 dB". Enter "OK" and you will see Fig 8, thus demonstrating that the filter programs calculations are correct.

Anyone wanting more details should investigate the two menus "Scales" and "Options", there are several options for altering the results diagram.

2.2. Second simulation example: low noise MMIC amplifier

The full circuit diagram for a GPS

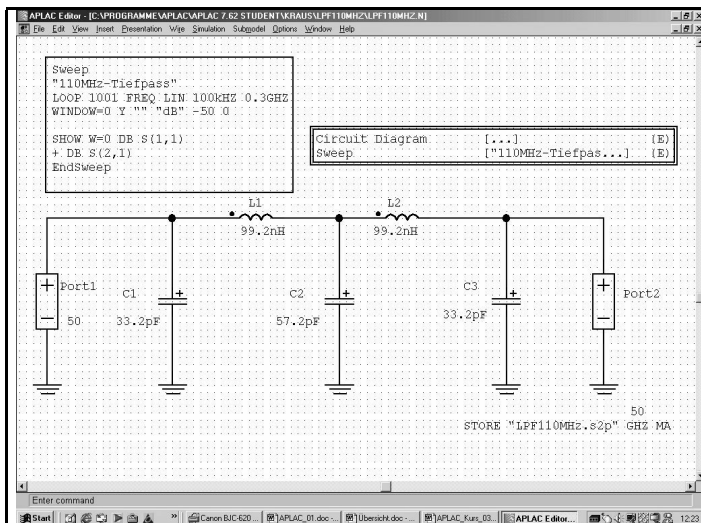


Fig 6: The completed settings for simulation of the 110MHz low pass filter.

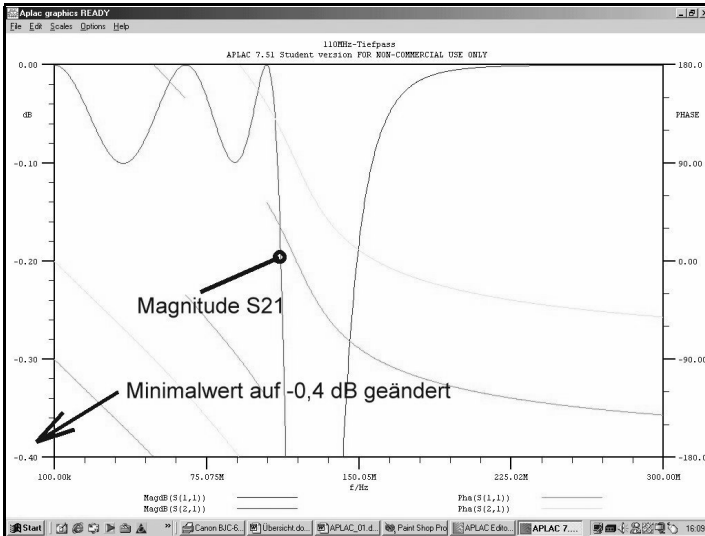


Fig 7: APLAC simulation results for the 110MHz low pass filter showing S11 and S22.

pre-amplifier (mean frequency: 1,575.42MHz) can be seen in Fig 9. The noise matching required has already been carried out by two microstrips at the input. The printed circuit board material is to be double sided Rogers 0.813mm thick (25 MIL). The underside forms a continuous earth surface, with the earth connections on the topside being connected by a sufficient number of feed-throughs.

Thus a brief APLAC task list for this simulation might be:

The behaviour of the circuit in the frequency range from 0.1 to 10GHz should be described through the S parameters and the curve for the noise factor, F (in dB). You should also check whether any stability problems can be expected with the circuit.

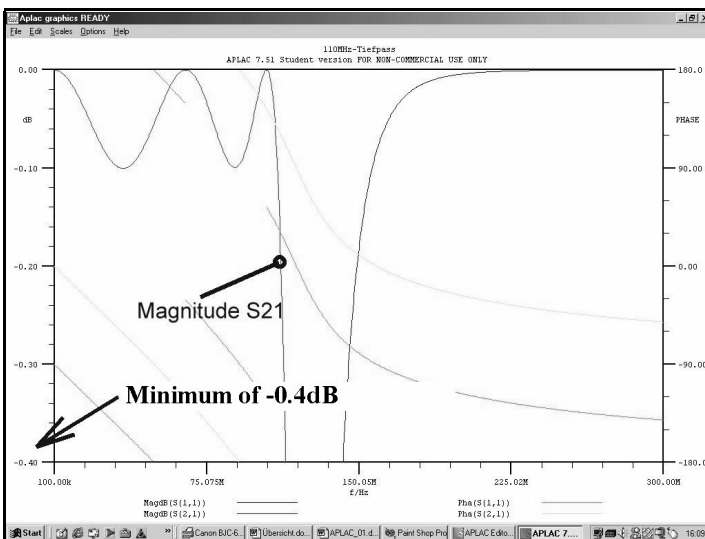


Fig 8: Repositioning the display clearly shows the Chebyshev ripples in the S21 curve.

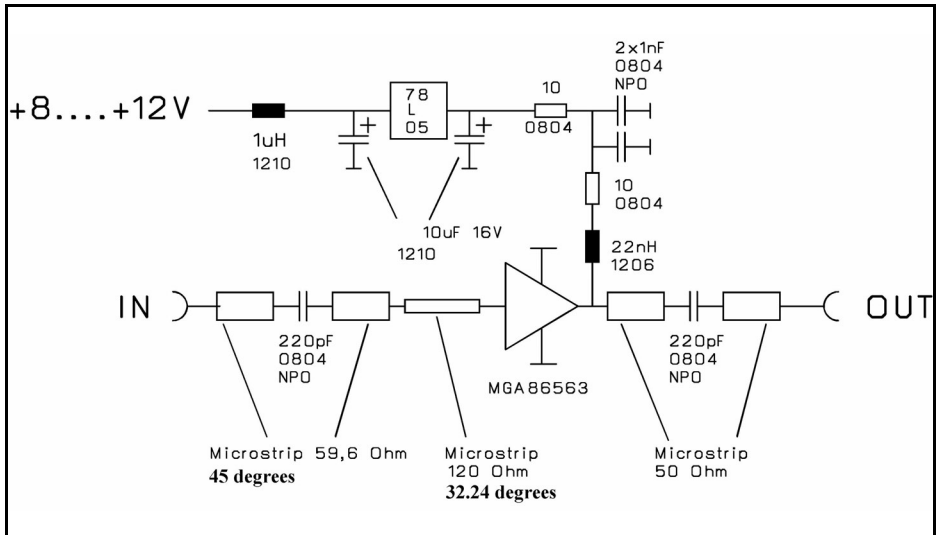


Fig 9: Circuit diagram of GPS pre-amplifier for simulation by APLAC.

2.2.1. Simulation screen

The screen required for the simulation can be seen in Fig 10, and some explanations are probably needed.

- The amplifier IC is represented by an “Nport” (Two Port) from the “macro library” that can be accessed by right clicking on the screen area.
- The S parameter file taken from the Agilent company's homepage, which also includes the noise data, is connected, through its attributes using “LOAD=m86563v5.s2p”. This file must be copied into the valid project file for our circuit.
- A “Prepare” object should be positioned in the object list box (“Prepare Noise”), which will prepare the noise simulation.
- The two microstrips used for the noise matching are first simulated as ideal lines which are “Tline” in the basic menu (right click mouse). The impedance has to be entered in the attributes. The electrical length is in wavelengths of the frequency, $FC=1,575\text{MHz}$ (for the first line, 90 degrees long, you need to enter “0.25”).
- On the output side, the supply voltage is fed to the MMIC through a small SMD 22nH choke (size: 1,206) and a 10 Ω SMD resistor. The self capacitance of the choke coil is also accounted for as 0.1pF (it can be calculated from the self resonant frequency specified in the data sheet) also 0.2pF for the capacitance of the solder pad linking the resistor and coil.
- The two earth connections for the MMIC must be connected to the lower earth surface using as many feed-throughs as possible. The inductance of these feed-throughs is usually the reason for any tendency to oscillate and any instability in such amplifier circuits, it must therefore be kept

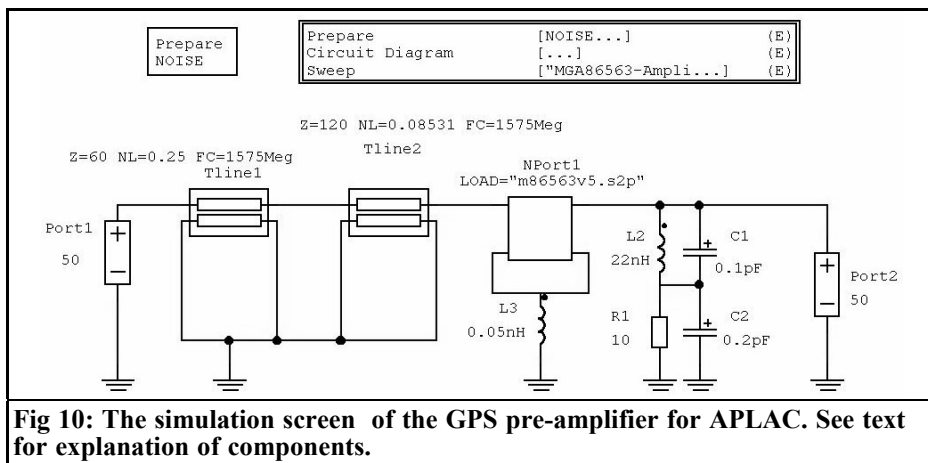


Fig 10: The simulation screen of the GPS pre-amplifier for APLAC. See text for explanation of components.

as low as possible! If we set a value of approximately 1nH per feed-through per 1mm of printed circuit board, it then follows that with 10 feed-throughs on each side, we can expect a total inductance of about 0.05nH. This inductance is shown as component L3 in the circuit, so that it can be used to monitor the stability.

- A 50Ω port is connected to the input and another to the output as terminations.

2.2.2. Sweep file to simulate S parameter and noise factor

This file (as always, as the last position in the Object Box List) must look like:

```
"MGA86563-Amplifier"
LOOP 1001 FREQ LIN 0.1GHZ 10GHZ
WINDOW=0 Y "" "dB" -40 0 GRID
WINDOW=1 Y "" "dB" 0 40 GRID
WINDOW=0 Y "" "dB" 0 4 GRID

SHOW W=0 Y MagdB(S(1,1))
+ Y MagdB(S(2,2))
+ W=1 Y MagdB(S(2,1))
+ W=2 Y NoiseFigure
+ Y MinNoiseFigure Marker=1
```

Explanation:

In the first half (before the blank line) we have first the circuit name "MGA86563-Amplifier" (it will be shown in every result following the simulation). This is followed by the requirement to simulate a maximum of 1001 points in the frequency range from 100MHz to 10GHz (maximum value for free student version).

In the following three diagrams (Window = 0 to Window = 2) a grid is programmed for the vertical axis instead of dB divisions, in order to make the values easier to read off. The display range in the first diagram (for S11 and S22!) goes from 40 to 0dB, whereas a range from 0 to + 40dB makes more sense in Window 1 for S21. A scale from 0 to 4dB is the best choice for the noise factor. Now (never forget this!) a blank line is required.

The subsequent display notes for the diagrams are probably self explanatory. Window 2 is programmed to display both the noise factor actually obtained and the theoretical minimum possible (NFopt). Finally, press <Control + S> on the keyboard to start the simulation.

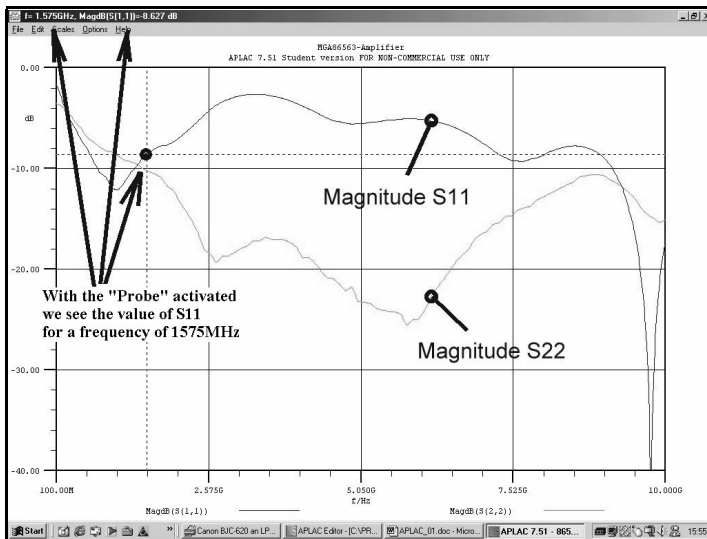


Fig 11:
Simulation results
for the GPS
pre-amplifier
showing S11 and
S22 up to 10GHz.

2.2.3. First simulation results

Following some calculation time, the results appear on the screen. Click to select the desired window into the foreground and zoom. The values for the input reflection, S11, and the output reflection, S22, are shown in Fig 11. The curve for S22, in particular, gives no cause for complaint, and only gets better as the frequency becomes higher. S11, on the other hand, is determined by the noise matching circuit and must be accepted as it is. Under “Options”, you will find the option “Probe”, using this you can go to each point of a curve and the associated value will be shown in the top left hand corner of the screen.

In the S21 curve, shown in Fig 12, the only cause for complaint might be that the maximum amplification is at a higher frequency than ideal. We get a very nice result from Fig 13, where the noise factor, between 1GHz and 2.5GHz, is very close to the minimum value

2.2.4. Stability control

2.2.4.1. Basic requirements for stability

The best circuit will be of no use if it has a tendency to uncontrollable self excitation, i.e. if it oscillates after assembly. In the specialist literature and in practice, various checks are standard, which all make use of the S parameters of the circuit using specific guidelines.

The oldest and best known method will now be examined in greater detail. It operates using the stability factors “K” and “B1”

These two variables are dependent only on the S parameters of the two port network used and not on the terminations. They can be used to express the condition for “absolute freedom from oscillation under all conditions and load and/or internal resistances” (unconditional stability) in the following manner:

$$K > 1 \text{ and } B1 > \text{zero}$$

The two variables are calculated from the

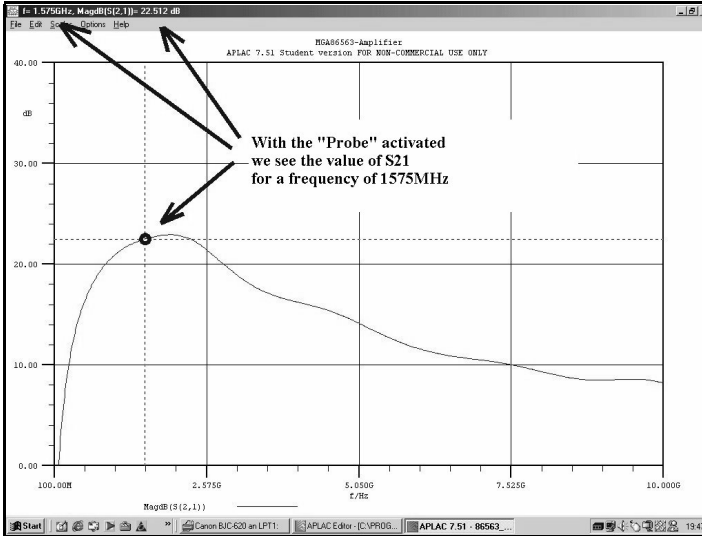


Fig 12:
Simulation results
for the GPS
pre-amplifier
showing S21 up
to 10GHz.

S parameters in the following manner (but we naturally leave this to a program...):

where:

$$|\Delta| = |S_{11} \cdot S_{22} - S_{12} \cdot S_{21}|$$

$$K = \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |\Delta|^2}{2|S_{12} \cdot S_{21}|}$$

and

$$B1 = 1 + |S_{11}|^2 - |S_{22}|^2 - |\Delta|^2$$

Note:

APLAC no longer uses this technique, but instead makes a total of three more modern but very different control functions available:

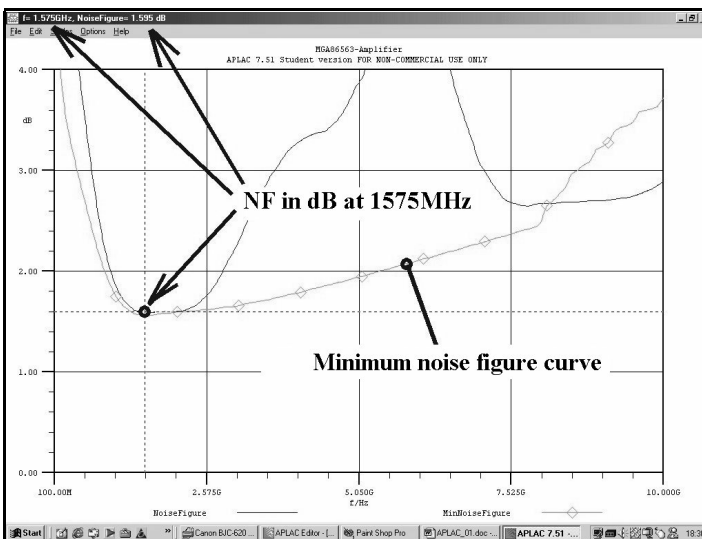


Fig 13:
Simulation results
for the GPS
pre-amplifier
showing noise
figure.



First option:

Following the method just discussed (and perhaps also because many people are still so used to it), the following variants can be used by calling up the “S_K” and “S_D” functions:

The stability factor K must exceed 1 AND the S matrix determinant $D = \Delta$ must be less than 1, if the circuit is to be stable overall (unconditionally stable).

Second option:

We can work with only a single stability factor “ μ ” and exercise control using the function “S_u”. Then we have:

The stability factor μ must exceed 1 if the circuit is to be “unconditionally stable” overall.

Third option:

This most modern solution would be the one to fulfil the “Nyquist criterion”. It is described as an “Advanced stability analysis” (see APLAC Users Guide, Volume 2, pp. 3-26) and represents the “standardised determinants function” in the complex plane, using the option “NDF”. It maintains that:

As soon as the origin (“zero point”) of this complex plane is not embraced by the NDF curve, the circuit remains stable overall.

But for this an analysis must first be carried out in the time domain e.g. using an AC source because results cannot be obtained by just using the S parameters. So this option has been omitted.

2.2.4.2. Stability control using K and Δ and also using μ

Methods 1 and 2 are combined for the amplifier project. Naturally, a few changes are needed in the sweep file for this purpose:

- In the simulation section, we need a third window for the three stability control factors. The graduation of the scale should go from 0 to 4.
- In the display section, we arrange that “D”, “K” and “ μ ” are not only displayed but are also marked differently.

This sweep file can be seen In Table 1 (the additions are marked in bold):

(Note: In this section of the sweep file, the change from the instruction “SHOW” to “DISPLAY” is deliberate. “DISPLAY” is the complete function for all types of representations that require more calculations to be carried out but also offer more options. “SHOW” is the rapid, slimmed down version with restricted capabilities, it is not adequate for the stability factors.

The result of this simulation is shown in Fig 14. In spite of the total feed-through inductance of 0.05nH transferred into the circuit as a precaution, the circuit remains stable over the entire frequency range (K and μ exceed overall, whereas D is less than 1 or 0.5).

2.2.4.3. Additional control using the stability circuits

The minute you get involved with circuits that have a tendency to oscillate only at specific internal resistances or loads (potentially unstable devices), the method described in the previous section, using the three factors K, μ and D, is no longer adequate. We know only that a tendency to oscillate can arise as soon as K or μ falls below 1 or D exceeds 1, but we do not know the critical conditions yet. More comprehensive analyses are needed here, which give a result in the form of two circles showing precisely these unstable areas of the associated reflection factors in the Smith diagram.

One circle shows how the stability, K,

**Table 1: Modified sweep file.**

```
"MGA86563-Amplifier"
LOOP 1001 FREQ LIN 0.1GHZ 10GHZ
WINDOW=0 Y "" "dB" -40 0 GRID
WINDOW=1 Y "" "dB" 0 40 GRID
WINDOW=0 Y "" "dB" 0 4 GRID
WINDOW=3 Y "" "" 0 4 GRID

SHOW W=0 Y MagdB(S(1,1))
+ Y MagdB(S(2,2))
+ W=1 Y MagdB(S(2,1))
+ W=2 Y NoiseFigure
+ Y MinNoiseFigure Marker=1
DISPLAY WINDOW 3 Y "D" S_D
+ Y "K" S_K Marker=1
+ Y "μ" S_u Marker=2
```

depends on the load (more correctly, on the associated reflection factor, Γ_{LOAD}), the associated technical expression is:

Stability K (Load Plane)

The other circle shows how the stability, K, depends on the internal resistance of the source (better: once again, on the associated reflection factor, Γ_{SOURCE}).

Here the technical expression is:

Stability K (Source Plane)

Yet both these "stability circles" are only valid for *a single frequency* and the simulation must be repeated if the frequency is changed.

Thus the circuit can often still be stable for a specific frequency if the internal resistance or load is known and constant, although the stability factors, K, μ and D are already warning of a potentially unstable situation.

A very simple rule applies for the evaluation of the circles:

- If both circles (for Γ_{SOURCE} or Γ_{LOAD}) run completely outside the Smith diagram, the circuit is guaranteed to be unconditionally stable.
- It is likewise unconditionally stable if the Smith diagram (for Γ_{SOURCE} or Γ_{LOAD}) is completely surrounded (i.e. enclosed) by a stability circle. That sounds a little crazy, but the background is as follows. The centre of the stability circle then lies in the infinite and its circular

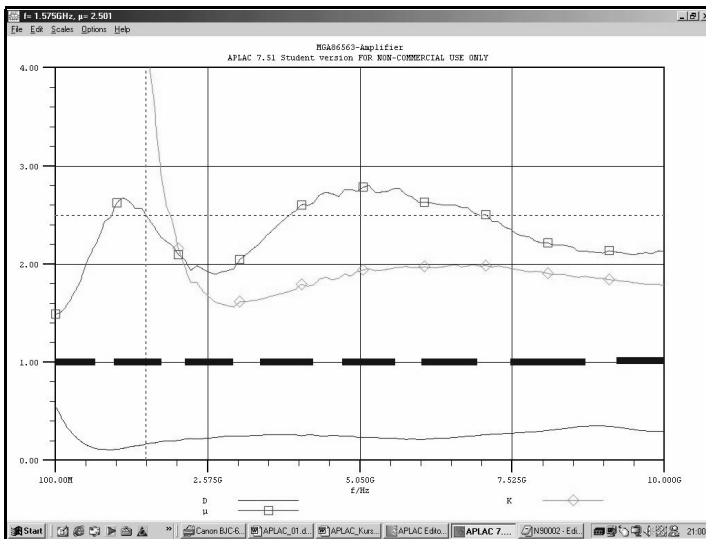


Fig 14:
Simulation results
for the GPS
pre-amplifier
showing stability.

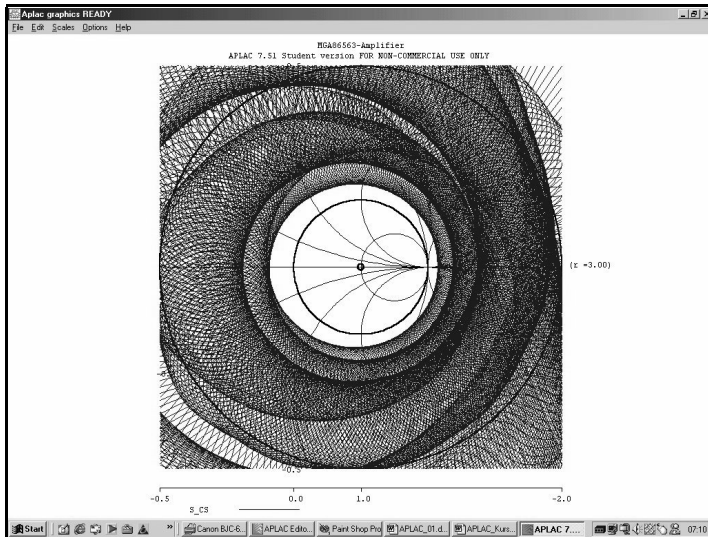


Fig 15:
Simulation results
for the GPS
pre-amplifier
showing the
smith diagram
for the source
side.

area covers the entire universe, but still leaves the Smith diagram free.

To help you visualise this, you can imagine a globe. Push a pin into Germany, which represents the centre of the Smith chart. Stick another pin into the other side of the globe e.g. in California, which represents the centre of the stability circle.

If we now continually increase the radius of the “Californian stability circle”, then it will finally approach the pin in Germany from all sides around the globe. Eventually it is pinned into a tiny little free area and that is the stable region expected, in which there is still no oscillation.

In both cases, “conventional analysis” simultaneously gives a value of $K > 1$, $\mu > 1$ and $D < 1$.

But if any parts of the circles are inside the Smith diagram, then the circuit oscillates if the internal resistance or load lies in the intersection zones. Then K is also < 1 or $\mu < 1$ or $D > 1$!

It is thus the developers task to check whether, in practical operation, these

critical resistances can occur. If not, then the circuit can be implemented by careful design and can function reliably.

APLAC is able to do the calculation of functions corresponding to these circles which we must call up. They are:

S_CS for the centre and S_RS for the radius of the source stability circle, together with S_CL for the centre and S_RL for the radius of the load stability circle.

And now here's an interesting thing, which is probably not something any of APLAC's competitors can offer:

Using “manual programming” of the APLAC sweep file, you can easily (e.g. by entering the following):

**LOOP 1 FREQ LIN 1.57542GHz
1.57542GHz**

have the ratios and all selected parameters displayed for a single individual frequency (here 1.57542GHz). But only APLAC will also calculate and show all the associated stability circles for a wider frequency range during sweeping!

All other products (e.g. APPCAD, Ansoft, etc.) are restricted here to a single selectable design frequency.

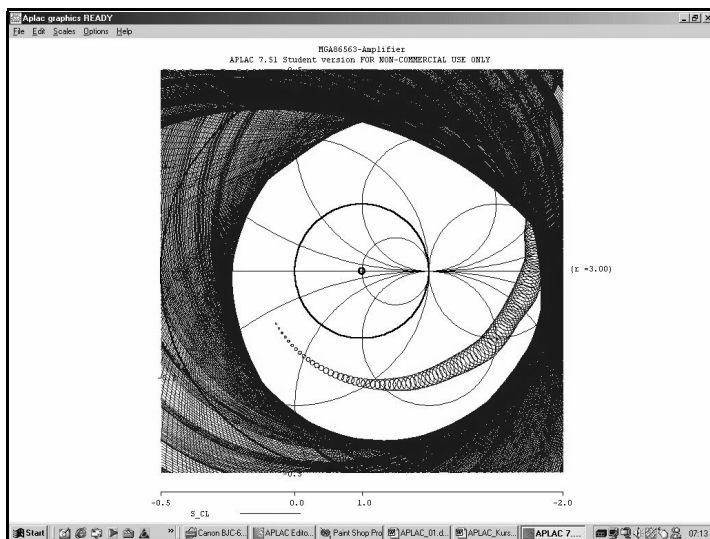


Fig 16:
Simulation results
for the GPS
pre-amplifier
showing the
smith diagram
for the load side.

As a final step in the amplifier project, this can be viewed. To do this, expand the sweep file again adding two Smith diagrams with the radius $R=3$:

The result of the work can be seen in Fig 15 for the source side, and in Fig 16 for the load side. It has already been established that the circuit is stable, but now we can see the safety margins for the different frequencies and can, if necessary, pick individual frequencies out, using "Probe", for more precise analyses.

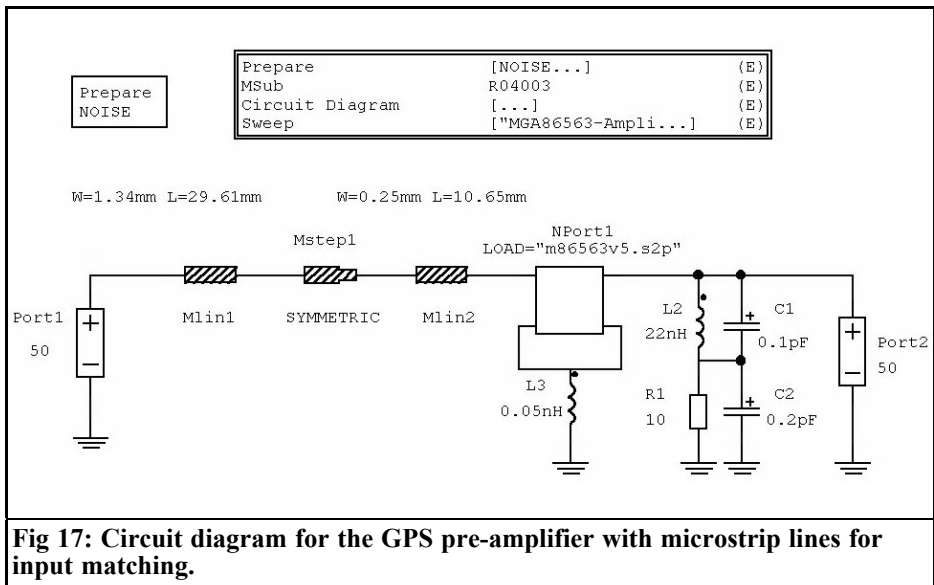
Nor would it be a problem to look at unstable regions and work out which source impedances or load impedances to avoid and then permit "restricted operation", e.g. with precise 50Ω loads.

That can become important for most PHEMTs that have been designed for use at 10GHz and almost always have a tendency to oscillate at low frequencies. We can use just such tricks for such projects as a super low noise Meteosat or 2 meter pre-amplifier with a noise figure of less than 0.5dB to permit clean operation.

2.3. Simulation using real microstrips

In conclusion, Fig. 17 shows the amplifier circuit, where the ideal transmission lines have been replaced at the input by two actual microstrips with different widths and (to correct the effects when lines with different widths meet together) a "symmetrical Mstep". Microstrips can be called up using: "right click mouse / Microwave / Microstrip / Mlin(stripline).

The circuit dimensions required were calculated using several programs for comparison. First using PUFF, then using the stripline calculator TRL85, then using the microstrip calculator from the new AppCAD, version 3.0, and finally using a self written APLAC microstrip calculator (APLAC merely prepares the necessary functions for this users have to compile the calculation instructions themselves). There were no significant differences. Likewise, the new amplifier simulations fully coincide with the values from the previous section.



3. Summary

APLAC offers an almost incredible number of options, but presents itself with a very plain and simple user interface. That means that, in comparison with other products, considerably more training is needed and routine use is decidedly harder work. Yet if we consider that all the simulations and investigations presented were undertaken exclusively using the free student version, things look much better. The author therefore uses this student version for the simulation of all those things which are not available in the familiar PUFF system, and often exchanges simulation results or partial solutions between the two programs in the form of S parameter files.

Even using the free version, there is no barrier to such things as impedance jumps or open ended extensions of microstrips, interdigital capacitors, bends, etc. Yet the message "Memory restricted in this version" often puts a

dampener on one's enthusiasm in such applications.

However, the author considers himself fortunate finally to have convinced his department to procure a full version under the APLAC university programme. As a consequence of this, a 120 page APLAC tutorial has been created in four sections:

- Part 1: Training + simulations in time domain and frequency range
- Part 2: Investigation of digital circuits
- Part 3: Passive microwave and microstrip circuits
- Part 4: Active microwave circuits

Part 1 has also been translated into English for an international introductory course in our school.

A copy of this "APLAC-CD" can be obtained from the author by email at cost price: krausg@elektronikschule.de